Gunthard Kraus, DG9GB

Modern Design for Band Pass Filters Made From Coupled Lines

Nowadays there are many aids available to any electronics developer. Even for development work in the area of high-frequency engineering, there is some very powerful software in existence, some of which is available without charge on the Internet.
The use of modern design and simulation tools is described below by means of examples.

1. Foreword

Some years have passed since the series of articles entitled "Design and realisation of microwave circuits" in [1], in which this subject was dealt with comprehensively. In the intervening period, the options for procuring information and for circuit simulation have multiplied greatly. In addition, the analysis options are more precise, thanks to continuous improvements in the CAD field.

A "test version" or "student version" of almost any modern CAD or simulation programs can now be obtained from the Internet, including the original manuals, which are usually complete textbooks in themselves - and mountains of application notes on almost any subject. The real problem thus becomes how to make a suitable choice. "Know-where and know-how" are also important, for all test versions of what are usually very expensive programs have some kind of limitations. And theres nothing more irritating than to slog away at familiarising yourself with a new program and then suddenly realise that the program available just cant go any further with the problem youre working on.

So the idea here is to demonstrate the correct and successful design of stripline band-pass filters, together with their implementation in practise. We shall compare not only the procedures but also the degrees of success, using the tried and tested CAD program "PUFF" (Version 2.1) and the ultra-modern student version of ANSOFT Serenade.

2. A glance at the technology

Band-pass filters serve to "separate out" a specific desired frequency range, while simultaneously suppressing, as far as possible, all undesirable signals outside this range. The following filter models can be considered for the microwave range in this context:

- Waveguide filters (for very high power levels)
- Coaxial filters

- Helix filters
- Filters made from ceramic resonators
- SAW filters
- Inter-digital band-pass filters
- Stripline filters with coplanar structures
- Microstrip filters made from coupled lines
- Hairpin filters, etc.

If we also lay down additional requirements, such as

- DIY manufacture as simple printed circuit board at lowest possible cost
- easily convertible to other frequencies without high costs or problems
- no smoothing
- absolute reproducibility

then the two last types are usually given preference. In this context, hairpin filters represent a modified version of the standard stripline filter for shortening the construction length and increasing the edge steepness. The disadvantages of larger dimensions must be taken into account here.

3. Principles of stripline band pass filters made from coupled lines

Here we are using "coupled lines", i.e. two striplines which are running in parallel and close together. Due to this running together closely in parallel, we obtain not only a capacitive coupling (via the electrical field) from one line to the other, but also a magnetic coupling. The magnetic field of one line induces an electrical voltage in the second line and thus transfers electrical power. The remarkable thing here is that the different waves triggered through this coupling from one line onto the other are added together in only one direction. But in the other direction they are in antiphase and try to cancel each other.

This is precisely the behaviour of a directional coupler and it is also the main applications area for this line structure. This behaviour can be used to separate forward and return waves in a system!

However, the description of such a component for simulation can be expanded further:

Due to the fact that some of the waves

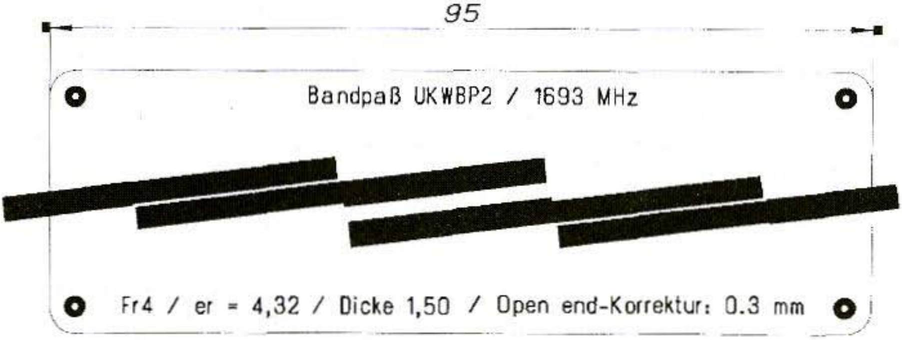


Fig 1: PCB layout for 16943 MHz Bandpass Filter

triggered in the second line are in-phase and some are anti-phase, it is necessary to specify two different impedance levels, namely

a) the EVEN impedance (or: in-phase impedance) and

b) the ODD impedance (or: anti-phase impedance).

The EVEN impedance level is always higher than the system impedance (usually 50 Ω), whereas the ODD impedance is essentially lower than the system impedance used.

The relationship of the three impedance levels to one another always depends on the formula

$$Z^2 = Z_{EVEN} \cdot Z_{ODD}$$

Note:
As soon as even one impedance is specified, many CAD programs react in the following way: if the impedance exceeds the system impedance, Z, then it is indicated as the EVEN impedance, and the missing ODD value is calculated in accordance with the above formula - and vice versa.

3.1 Now for stripline band-pass filters

If several such coupled line pairs, with 90 degrees of electrical length, are connected together in series, the line sections act as resonators and "the input signal is transmitted smoothly from input to output, only in the range around this frequency" in this way, the desired band-pass behaviour is obtained. Unfortunately, this is repeated at the odd multiples, i.e. for example the triple frequency, etc...

For the practical implementation, see Fig. 1.

It can easily be seen that, in addition to the three coupled line pairs, the 50-Ω striplines are also used as connection to the SMA sockets. The underside of the printed circuit board is a continuous earth surface.

4. The design path: from the standard low-pass to the stripline band-pass filter

4.1 Preliminary work

The circuit developer is initially faced with the following decisions:

Which type of filter is the correct one?
The choice will fall, for example, on Bessel, Butterworth, or Chebyshev filters. Basically:

Chebyshev filters display ripple in the transmission range, but as against that they can offer good edge steepness for the transition into the filter attenuation band.

If, in contrast, we need better group delay behaviour and no ripple in the transmission range, we go for Butterworth filters, though their edge steepness in the filter attenuation band is markedly lower than that of the Chebyshev type.

If the filter has to remain absolutely gentle and as smooth as possible at all points, that leaves only the Bessel filter. Mind you, we pay for this "gentle" behaviour with a very "tired" transition from the transmission range to the filter attenuation band (in order to keep phase distortion as low as possible). Thus, there is scarcely any "edge steepness" to speak of in the filter attenuation band.

Then comes the question of the degree of filtration, N, which for normal low-pass filters directly corresponds to the number of components needed. A greater degree of filtration brings about steeper edges in the filter attenuation band, but in practise

the attenuation in the transmission range is also increased, due to the greater number of components and their losses.

In practise, the type of filter which is very frequently used is the Chebyshev, with N between 3 and 5. For this reason, a filter from this group is taken as an example here.

The next decision relates to the system impedance (usually 50 Ω). Moreover, especially for Chebyshev filters, the maximum passband ripple, the reflection factor, etc., must be determined.

It should be borne in mind that the variables:

- passband ripple (oscillations of S21 and / or the transmission loss)
- reflection factor r
- voltage standing wave ratio VSWR
- S11
- reflection attenuation ar

are inseparably associated with each other in the Chebyshev type! The following relationships apply here:

a) Between the reflection factor r and the voltage standing wave ratio VSWR:

$$r = \frac{VSWR - 1}{VSWR + 1}$$

b) Between the reflection factor r and the passband ripple (maximum transmission loss in dB)

$$a_{MAX} = 10 \cdot \log \frac{1}{1 - |r|^2}$$

c) Between S11 and / or S22, the reflection factor r and the reflection attenuation a_R:

With correct matching, S11 and / or S22 correspond precisely to the reflection factor of the filter, but are normally specified in dB:

$$S11 = S22 = 20 \cdot \log |r|$$

The reflection attenuation is then simply the "negative dB value of S11 or S22"! Correctly:

$$a_r = 20 \cdot \log \frac{1}{|r|}$$

The following summary table (drawn up in accordance with the above formula) is intended to serve as a small design aid:

Reflection factor |r|	Reflection attenuation a_R	S11 or S22	Chebyshev Ripple of Trans loss
50 %	6dB	6dB	1.25dB
20 %	14dB	14dB	0.177dB
10 %	20dB	-20dB	0.0436dB
5 %	26dB	-26dB	0.01dB
2 %	34dB	-34dB	0.0017dB
1 %	40dB	-40dB	0.00043dB
0.5 %	46dB	-46dB	0.0001dB

(In practise, a maximum reflection factor between 5 and 10 % is usually sufficient...)

4.2. Designing a GPS bandpass filter

A bandpass for GPS with the following data is intended to serve as a design example:

Filter type	Chebyshev
Mean frequency	f_0 = 1575 MHz
Lower limiting frequency	f_{Min}=1550 MHz as ripple limiting frequency
Upper limiting frequency	f_{Max}=1600 MHz as ripple limiting frequency
Degree of filtration	N = 3
System impedance	Z = 50 Ω
Max. reflection factor	|r| = 10 %
Max. ripple	a_{max} = 0.0436 dB
Reflection attenuation in transmission range	ar = 20 dB

S11 in transmission range S11 = -20 dB
if at all possible, less than

The selected reflection factor r = 10% gives a maximum ripple of 0.0436 dB in the transmission range.

This means that S21 can fall as low as -0.0436 dB, whilst S11 and S22 never exceed 20 dB.

Note:
The following calculation path is taken from the book "Microwave Engineering" by David Pozar [5], Page 484.

Additional note:
The degree of filtration, N, should always be selected to be odd (i.e. 3, 5, 7...), because only then are the source resistance and the load resistance identical. Apart from this, make sure that the number of coupled line pairs is always 1 more than the selected degree of filtration. For N=3, there must thus be four line pairs in the layout.

And now to the individual design steps:

1st step:
First we need the filter coefficients of a single low-pass filter for this case. For this we can, for example, use the "faisyn" program (obtainable, for example, from http://www.rfglobalnet.com).

The above filter data are entered in succession when the program makes the corresponding requests, and the option "Parallel Capacitor" is selected. Thus the following table is finally obtained, with the 4 coefficients required for the calculation (Fig. 2):

g1 = cap1 = 0.8532

g2 = Ind1 = 1.1038

g3 = g1 = cap = 0.8532

g4 = normalized load resistance = 1.0

2nd step:
Specification of fractional bandwith of pass:

$$\Delta = \frac{f_{max} - f_{max}}{f_{mean}} = \frac{1600-1550}{1575} = \frac{50}{1575} = 0.031746$$

3rd step:
Now we come to the admittance inverter constants for the four line pairs:

1st line pair:

$$Z_0 \cdot J_1 = \sqrt{\frac{\pi \cdot \Delta}{2 \cdot g_1}} = \sqrt{\frac{\pi \cdot 0.031746}{2 \cdot 0.8532}} = 0.24175$$

2nd line pair:

$$Z_0 \cdot J_2 = \frac{\pi \cdot \Delta}{2 \cdot \sqrt{g_1 \cdot g_2}} = \frac{\pi \cdot 0.031746}{2 \cdot \sqrt{0.8523 \cdot 1.1038}} = 0.05138$$

3rd line pair:

$$Z_0 \cdot J_3 = \frac{\pi \cdot \Delta}{2 \cdot \sqrt{g_2 \cdot g_3}} = \frac{\pi \cdot 0.031746}{2 \cdot \sqrt{1.1038 \cdot 0.8532}} = Z_0 \cdot J_2 = 0.05138$$

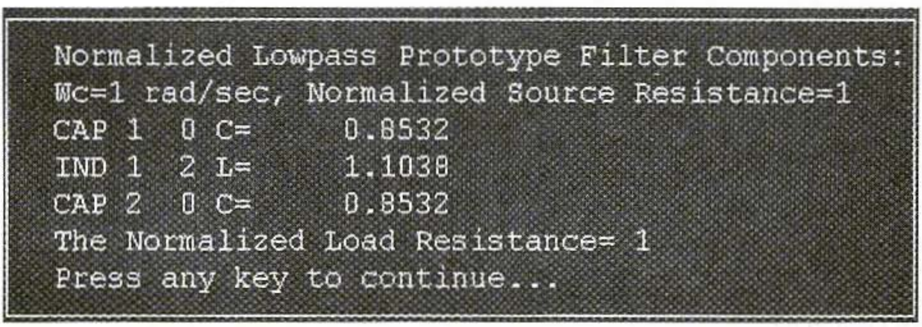


Fig 2: Filter Coefficients fFrom "Faisyn"

4th line pair:

$$Z_0 \cdot J_4 = \sqrt{\frac{\pi \cdot \Delta}{2 \cdot g_3 \cdot g_4}} = \sqrt{\frac{\pi \cdot 0.031746}{2 \cdot 0.8532 \cdot 1.00}} = Z_0 \cdot J_1 = 0.24175$$

4th step:

The EVEN and ODD impedances of a line pair are generally specified in accordance with the following formulae:

$$Z_{EVEN} = 50\Omega \cdot \left[1 + Z_0 \cdot J_N + (Z_0 \cdot J_N)^2\right]$$

$$Z_{ODD} = 50\Omega \cdot \left[1 - Z_0 \cdot J_N + (Z_0 \cdot J_N)^2\right]$$

For the first and fourth line pairs we obtain:

$$Z_{EVEN} = 50\Omega \cdot \left[1 + 0.24175 + 0.24175^2\right] = 65\Omega$$

$$Z_{ODD} = 50\Omega \cdot \left[1 - 0.24175 + 0.24175^2\right] = 40.8\Omega$$

For the second and third line pairs the values are:

$$Z_{EVEN} = 50\Omega \cdot \left[1 + 0.05138 + 0.05138^2\right] = 52.7\Omega$$

$$Z_{ODD} = 50\Omega \cdot \left[1 - 0.05138 + 0.05138^2\right] = 47.56\Omega$$

5. Use of PUFF

5.1. Simulation of ideal circuit using PUFF

First start "PUFF" And load the SETUP file. Then press the "F4" key and enter the following values for the Rogers material R04003:

with thickness 0.032

for impedance level zd = 50 Ω

the design frequency fd = 1575 MHz

the dielectric constant εr = 3.38

board thickness h = 0.813 mm.

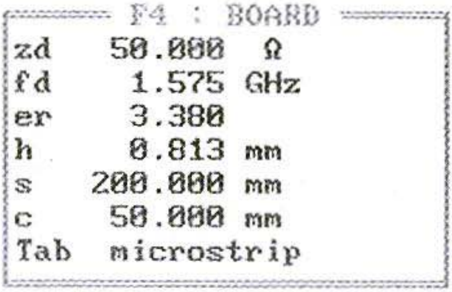


Fig 3: Starting Parameters

The printed circuit boards size "s" should be 200 mm, and 50 mm. is a sufficient distance between the connections (Fig. 3).

Then move into field "F3" and successively enter there the data for the two coupled line pairs required. Please transfer them precisely as shown in Fig. 4!

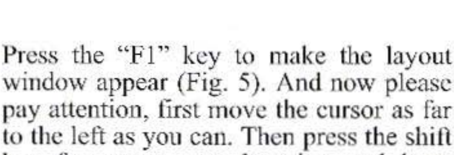


Fig 4: Data For Successive Line Pairs

Press the "F1" key to make the layout window appear (Fig. 5). And now please pay attention, first move the cursor as far to the left as you can. Then press the shift key for upper case lettering and keep pressing the "Cursor Left" key until you get to the desired location. Press "1" and port 1 is connected immediately.

Press letter "a" on the keyboard, followed by "Cursor Right". This positions the first line pair. Then press "b" and next "Cursor down", which connects up line pair "b".

Now press "Cursor down" again and the third pair is already sitting there on the screen. Finally press "a", "Cursor down"

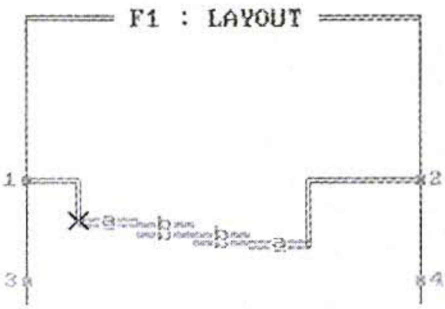


Fig 5: PUFF Layout Window

again and then "2" and port 2 is likewise connected to the exit of the circuit.

Use "F2" to go to the simulation window. Using "Cursor Up2 or "Cursor Down", you can move, not just in the top left-hand "Plot window", but also along the axes of a diagram in the bottom right-hand corner ("linear plot") (Fig. 6).

Here you pre-set, for example:

500 simulation points

Smith radius = 1

Representaiton of S11 and S22

Horizontal scales in the ratio: 1.5 ...1.7 GHz

Vertical scales in the ratio: -1.00...0 dB

And now please press "p", and you can watch the computer at work. If you think thats too slow, press "Q", and then all the calculations are done and the image is built up off-screen, and things go considerably faster!

If you look at the result now, youll undoubtedly be a little disappointed: its nowhere near a ripple with maximum 0.0436 dB; its bigger by a factor of more

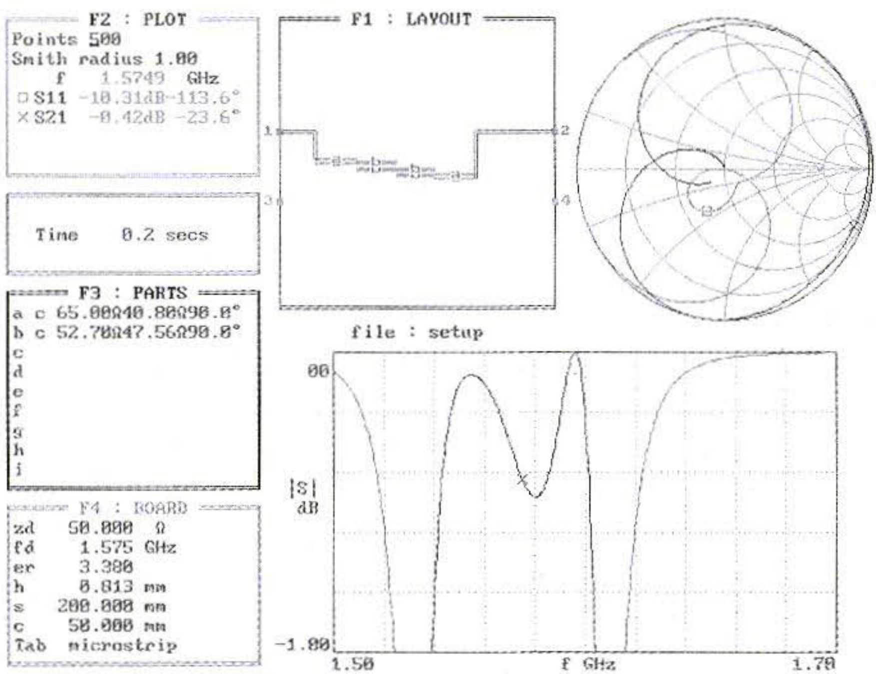


Fig 6: PUFF Simulation Window

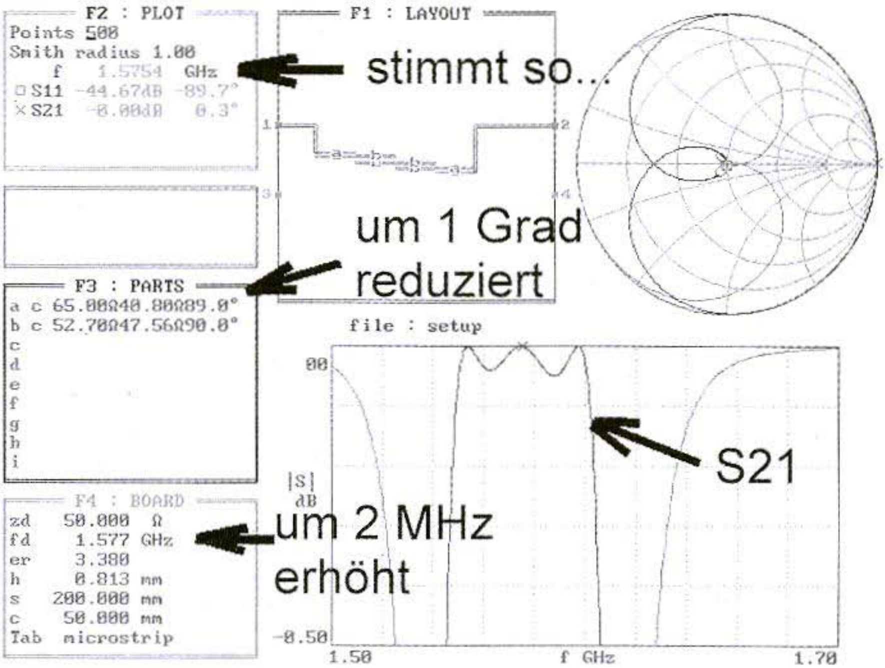


Fig 7: PUFF Simulation After Slight Corrections

than 10 and the value is 0.5 dB. But dont worry, we can still get round that. We just need to make some very slight corrections to the line data!

We need only reduce the electrical length of the first and fourth line pairs (part "a" in the parts list) by approximately 1 degree and increase the design frequency by 2 MHz in order to obtain the theoretical curve (Fig. 7)!

5.2. Simulation of the real circuit with PUFF

First call up the "SETUP.PUF" file from the PUFF directory into a text editor and then enter the remaining printed circuit board data (thickness of copper layer = 35 micrometres and surface roughness for a strip conductor polished until it gleams with grits and grinds or scouring powder approximately 2 micrometres. Loss factor "lt" of board material R04003 at this frequency max. 0.001):

....

zd	50,000 Ohms	{normalizing impedance. 0<zd}
fd	1.575 GHz	{design frequency. 0<fd}
er	3.380	{dielectric constant. er>0}
h	0.813 mm	{dielectric thickness. h>0}
s	200.000 mm	{circuit-board side length. s>0}
c	100.000 mm	{connector separation. c>=0}
r	0.010 mm	{circuit resolution, r>0, use Um for micrometres}
a	0.000 mm	{artwork width correction.}
mt	0.035 mm	{metal thickness, use Um for micrometres.}
sr	2.000 Um	{metal surface roughness, use Um for micrometres.}
lt	1.0E-0003	{dielectric loss tangent.}

....

The amended setup file is loaded back into PUFF and then the exclamation mark for each pair of coupled lines is entered in field "F3". This switches to "realistic modeling with all side-effects" (Fig. 8).

If we now place the cursor in field "F3" on part "a" and enter the equals sign, the actual data of the coupled lines are immediately faded into the dialogue field (Fig. 9).

Now we have to keep changing the values entered under "a" until the data determined in the preceding chapter are displayed in the dialogue field:

Ze = 65 Ω

```
==== F3 : PARTS ====
a c!65.00Ω40.80Ω90.0°
b c!52.70Ω47.56Ω90.0°
```

Fig 8: Switch To Realistic Modeling

Zo = 40.8 Ω

electrical length l = 89 degrees

It can be seen that to bring this about the entry for "a" has to be

c! 66.1 Ω 43.65 Ω 49.6°

This procedure must be repeated for part "b" (Fig. 10). The target is to obtain this display in the dialogue field:

Ze = 52.7 Ω

Zo = 47.56 Ω

l = 90 Grad

For this, finally, we need the entry:

c! 55.6 Ω 49.26 Ω 90.7°

Fig. 11 shows the result of the circuit simulation if the losses are taken into account.

The design frequency continues to remain at 1577 MHz, but following the simulation use <Page Down> to move the display cursor to 1575 MHz. We now have a transmission loss of approximately 2.5 dB.

If we correspondingly switch the value range in the two axes of the lower diagram, we can take another look at the long-range selection, i.e. the behaviour in the range between 1 and 2 GHz (Fig. 12).

5.3. Determination of mechanical, uncorrected line data with PUFF

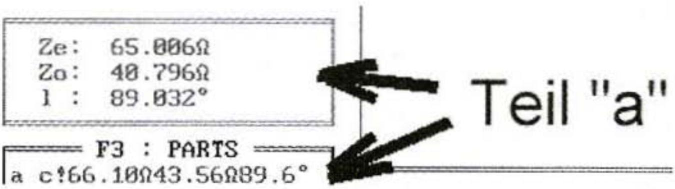


Fig 9: Data For Part "a"

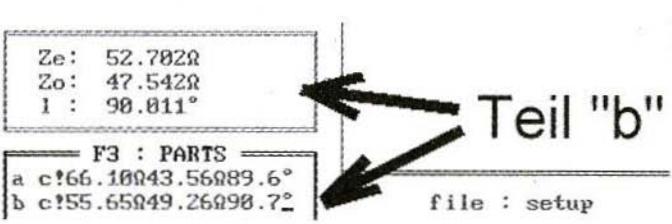


Fig 10: Data For Part "b"

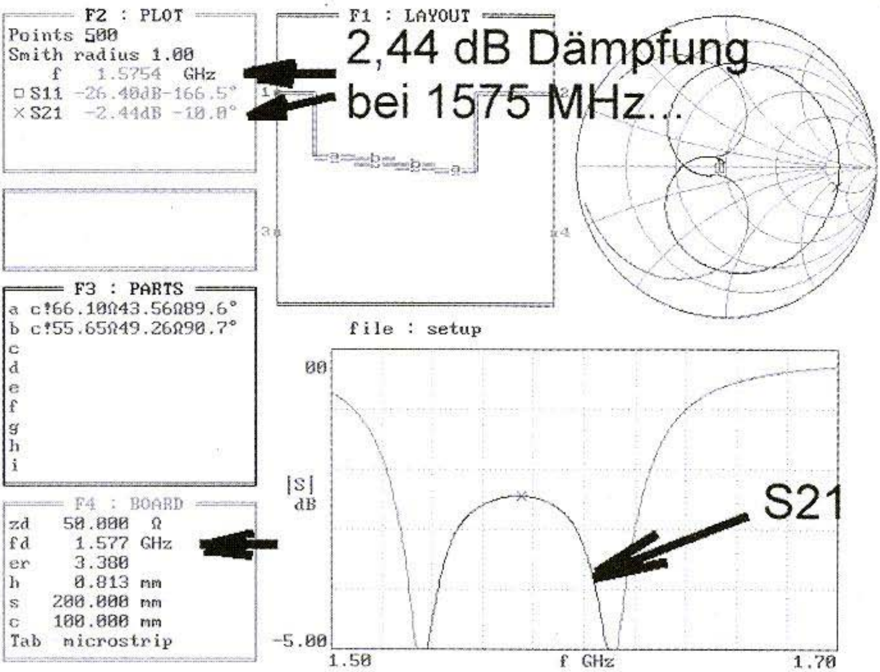


Fig 11: Results With Losses Taken Into Account

In order to get at the dimensions of the coupled lines, we move back again into field "F3" and delete the call-up sign at part "a" behind the letter "c" (for coupled lines). As soon as we key in the equals sign behind here, we obtain the desired values in the dialogue field (Fig. 13):

Length l = 29.34 mm

Width w = 1.59 mm

Interaction gap s = 0.31 mm

Repeat this for part "b", i.e. the two central pairs of coupled lines, and we correspondingly obtain Fig. 14:

Length l = 29.15 mm

Width w = 1.82 mm

Interaction gap s = 1.84 mm

Then, as a preliminary to the board design, determine the width of the 50 Ω feed likewise in the same way. It is modelled as "lossy transmission line with 90 degree length" and, as part "c", supplies a required width, w = 1.84 mm, following the deletion of the exclamation mark (Fig 15).

5.4. Necessary layout corrections

Here we are dealing either with striplines open at the end or with the meeting of two striplines which are of different widths. In both cases, the familiar OPEN-END correction is required, due to fringing, but one peculiarity should be taken into account here at the open ends of coupled lines:

The two line pairs are coupled to each other both electrically and magnetically. It is certainly true that the electrical field lines project beyond the open ends of the lines (so we need to do some shorten-

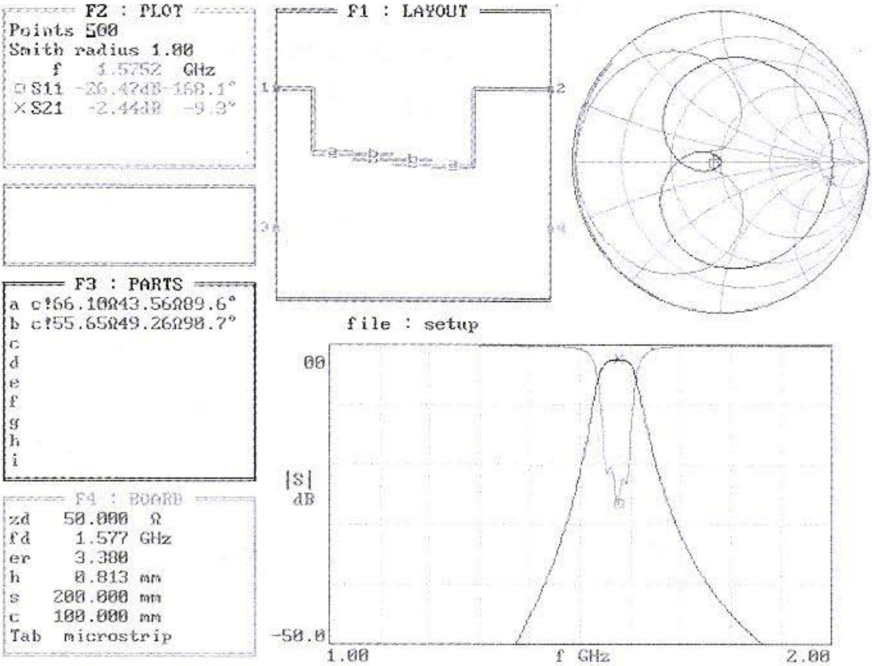


Fig 12: Simulation Results In The Range 1 - 2 GHz

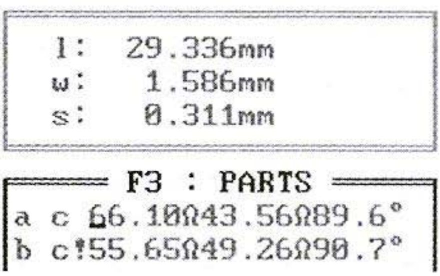


Fig 13: Results For Line Pair "a"

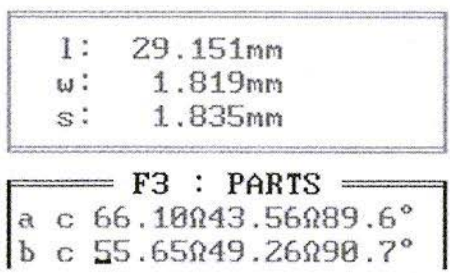


Fig 14: Results For Line Pair "b"

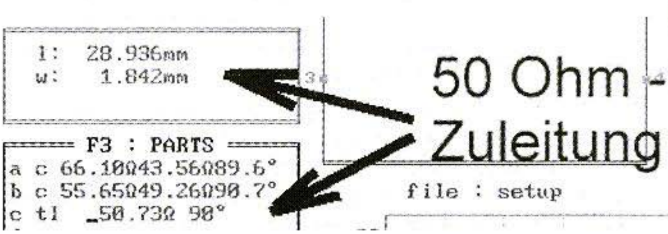


Fig 15: Determining The Width Of 50Ω Line

ing....), but the magnetic coupling decreases linearly in this area right down to zero.

For this reason, calculate in only half the "OPEN-END-EXTENSION" which would otherwise be normal and shorten the line correspondingly!

Apart from this, we now need several tools to create the layout:

a) The well-known, tried and trusted diagram for determining the OPEN-END-EXTENSION from the PUFF manual (Fig. 16).

b) A simple hand-drawn sketch (Fig. 17) with the electrical data of the individual line pairs already determined. Enter the necessary corrections.

c) A printed circuit board CAD program, which simultaneously makes it possible to solve tricky construction problems (e.g. GEDDY-CAD, tried and trusted for such microwave tasks for many years).

Procedure

1st step:

The first and fourth line pairs consist of two striplines each with a width of 1.59 mm. With the help of PUFF we obtain the impedance level for the pre-set printed circuit board data:

The result gives us: Z = 54.6 Ω

So we go to the above diagram from the PUFF manual. For this task, it supplies (with εr = 3.38) an open-end extension Δl/h of approximately 0.45. So these section pieces must be reduced by half of 0.45 x 0.813 mm = 0.18 mm at all open ends!

2nd step:

At the start and end of the bandpass, the 50-Ω feed is connected up with a width of 1.84 mm and turns into the (narrower!) stripline with a width of 1.59 mm. So the narrower line must be extended by a little piece measuring
(1 1.59 mm/1.84 mm) x 0.45 x 0.813 mm = 0.05 mm.

3rd step:

The two central line pairs have conductor widths of 1.82 mm. The associated impedance level (according to PUFF) is 50.4 Ω and at εr = 3.38 requires an

38 *Puff*

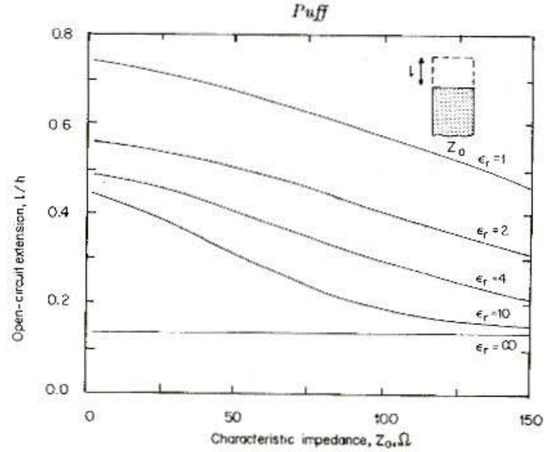


Figure 7.2 The open-circuit end correction in microstrip, plotted from (7.2). The artwork length correction in a parts list should be *negative*.

Fig 16 Diagram For Determining The Open End Extension From The PUFF Manual

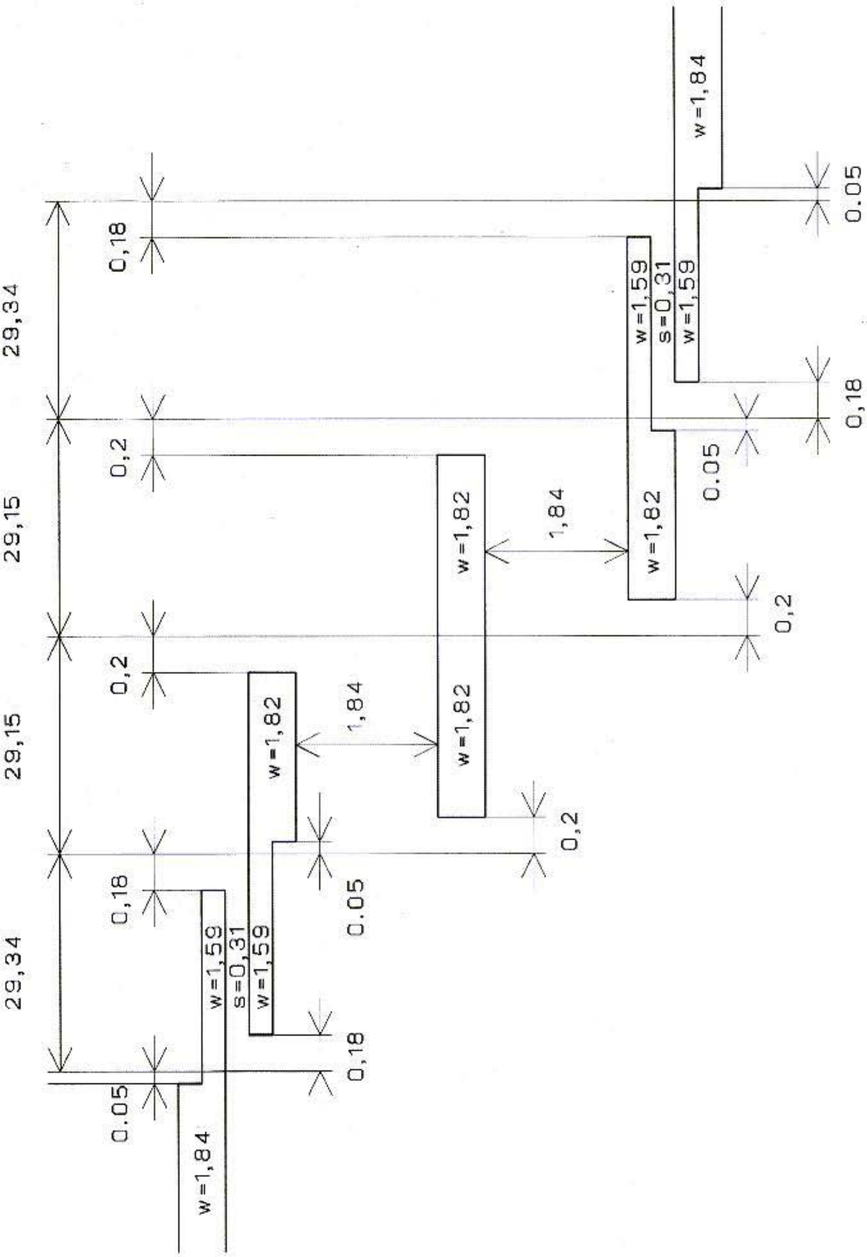


Fig 17: Sketch For The Filter

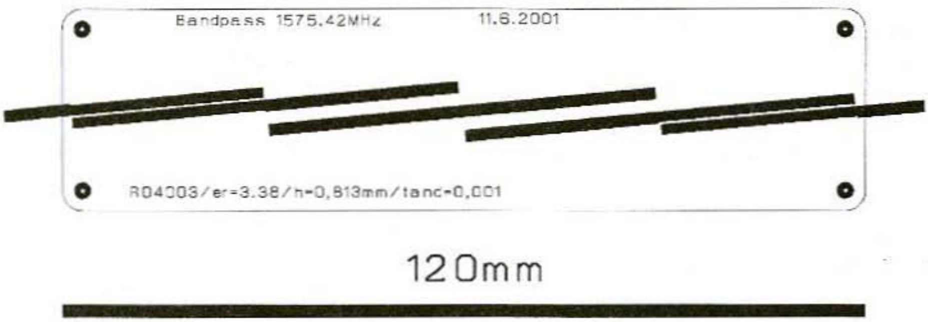


Fig 18: The Finished Layout

OPEN- END correction of 0.48 x 0.813 mm = 0.39 mm.

Again, only half of this value, i.e. approximately 0.2 mm, needs to be cut off from the two ends.

4th step:

When the first line pair meets the second and the third meets the fourth, there is a correction of

(1 1.59 mm / 1.82 mm) x 0.48 x 0.813 mm = 0.05 mm.

The wider line must be shortened and the narrower line must be lengthened by this amount.

You should never omit entering all these details on a sketch out of laziness. It is an obligatory stage in the layout design (Fig. 17) and you need to take considerable trouble just to produce the drawing. But this is as nothing in comparison with the time and expense which will be wasted if the behaviour of the prototype produced inexplicably displays big departures from the simulation and you laboriously have to check every individual dimension on the completed printed circuit board. Its really very annoying if a gross error in the board layout turns out to be the reason for this.

Normally if youve followed all the instructions in this article the divergences between the simulation and prototype are max. 1 - 2%.

The finished layout, prepared for incorporation into a milled aluminium trough, the internal dimensions of which are30 x 120 mm, is shown in Fig. 18.

One more tip for those who dont know what to do with the thick 120 mm long line under the printed circuit board: we need this to assemble the board equipment, unless there is a photo-plotter available, which can be used to bring us back to the correct dimensions by means of photography. Only in this way will we again find the correct scale and be able to handle the manufacturing problem posed by the narrow interaction gap for line pairs 1 and 4.

5.5. Use of TRL85 Stripline Calculator

To determine the data for the incorporation of the circuit into a screening housing, the "TRL85" stripline calculator from Ansoft is used.

Ansoft are known for supplying very high quality and expensive HF-CAD programs, but have always also had their eye on instruction and training! So on their Homepage on the Internet () we find a

student version of the microwave CAD package "Serenade" which can be downloaded free.

Following installation it will be recognised that a very good stripline calculator has been filed in a separate directory as "TRL85.EXE-file" together with excellent ONLINE help. It can be used separately at any time, copied down and transferred to other computers. You will very soon learn to appreciate its WINDOWS user interface.

If we now compare the simulation results from "TRL85" with the values from PUFF, we can make the following statements:

a) In normal cases, the data determined by PUFF and TRL85 for single and coupled striplines are practically identical.

b) In addition to this, TRL85 offers the advantage that screening can be brought into the simulation in the form of the "Cover Height" (the distance between the cover and the board).

c) With TRL85 all data which is of interest (impedance level, losses, broken down into dielectric loss, conductor loss and total loss... etc.) can be determined directly for a specific design frequency and given out. Unfortunately, PUFF cant provide for that, as there, though you certainly have to carry out calculations using these values, they are not displayed.

The "TRL85" program is presented in this issue in the section headed "The interesting program".

In Table 1 below, we now contrast the TRL85 microstrip simulations for operation without and with screening.

A comparison with the values determined by PUFF and used in preceding chapters shows that it is high time for a printed circuit board and correct dimensions.

Another tip:

TRL85, unfortunately, wont automatically apply the OPEN-END correction either. So again we have to fall back on the use of the diagram from the PUFF manual when a line end is hanging in the air somewhere.

Bandpass Specification	**TRL85 without screening**	**TRL85 with screening Distance = 13mm**
50 Ω Microstrip	Width = 1.84 mm	Width = 1.84 mm
First Line Pair	Ladder Width = 1.58321 mm	Ladder Width = 1.5769 mm
E=89, 1575.42 MHz	Spacing = 0.33225 mm	Spacing = 0.32877 mm
Z_e=65Ω, Z_0=40.8Ω	Length = 29.2968 mm	Length = 29.3133 mm
Second Line Pair	Ladder Width = 1.8216 mm	Ladder Width = 1.81525 mm
Z_e=52.7Ω, Z_0=47.56Ω	Spacing = 1.87272 mm	Spacing = 1.82327 mm
E=90, 1575.42 MHz	Length = 29.186 mm	Length = 29.2009 mm

Table 1: TRL85 Simulations With and Without Screening

6. Repetition of design using ANSOFT-SERENADE

6.1. Simulation of ideal electrical circuit

As "SERENADE" is already installed on the PC, on account of the "TRL85" stripline calculator, we can carry out a repeat operation and see how such a circuit is investigated by this very modern program. Naturally, we are interested, above all, in what improvements it can give us, in terms of ease of operation or precision.

Procedure

Launch the SERENADE software and start a new project (e.g. "BP1575_1").

Then look for the "ideal coupled line" (Fig. 19), position it on the screen four times and each time (see preceding chapter!) enter the EVEN and ODD impedances, the electrical length of 90 degrees and the operating frequency applying for this, 1575.42 MHz, in the "Property Editor". When the component is positioned, the editor opens automatically. If it does not, just double click on the left-hand mouse button on the circuit icon in the wiring diagram. Our ports are connected up at the input and output, but thats not enough yet!

The "Harmonica" circuit simulator is the problem now that two of the four connections have simply remained open for each coupled line pair, this is prohibited.

We could now apply a very high-ohmic resistance (e. g. 10 Mega-Ohms) to such an open connection. However "ideal line pieces working without load at the end (Stubs)" are considerably better and have lower losses, with an electrical length of zero at 1575 MHz and with the impedance level Z = 50 Ω. They cause no additional losses. Nor do they alter the data for the coupled lines.

Then the frequency block is re-set and the range between 1 GHz and 2 GHz is represented in 1 MHz steps. Rogers R04003 material is again used as substrate, with a thickness of only 32 MIL (0.813 mm), as this gives the filter structure smaller dimensions. The other data are as follows:

Dielectric constant	$\varepsilon_R = 3.38$
Metallisation Met1	Copper with thickness 35 µm
Surface roughness	RGH = 2 µm
Loss factor	TAND 0.001.

The entire circuit, as used for simulation, is shown in Fig. 20. Even for those who dont know the program yet, the components just produced can easily be identified. Frequency and substrate control blocks certainly need no further explanation.

Following a precise check of the circuit, the simulation can begin (button with gears) and the Report Editor can be activated (turquoise / grey button). Se-

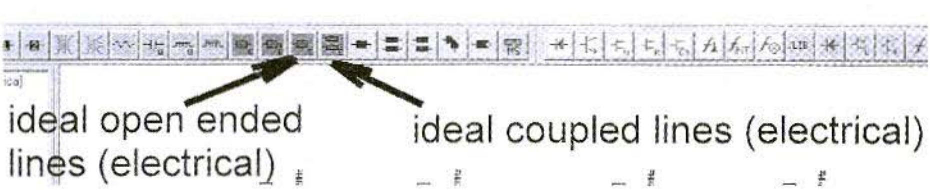


Fig 19: Toolbar For Serenade

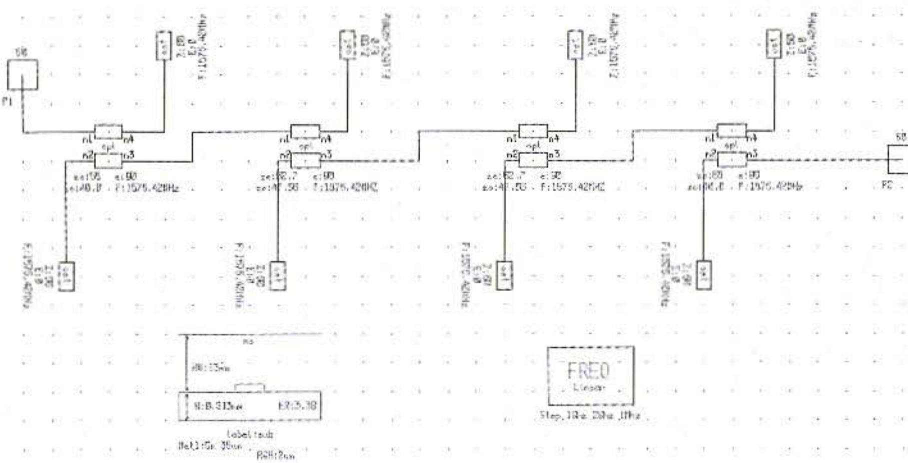


Fig 20: Circuit Used For Serenade Simulation

lect, for example, S11 and S21 in dB representation and examine the result (Fig. 21).

The result looks promising and S11 is never worse than the intended value in the passband 20 dB.

Use the right-hand mouse button and click on "ZOOM IN" repeatedly to bring out the precise sequence of S21 in the range from 0 to 0.1 dB between 1550 and 1600 MHz.

Only in this way can we assess whether the design path from the preceding chapter really supplies the correct values desired.

Fig. 22 shows a perfect and well-formed filter curve.

In practice, both the mean frequency (1575 MHz) and the minimum ripple value (0.041 dB) are in accordance with the design pre-settings.

6.2. Simulation of physical circuit using HARMONICA

Apart from simulating the bandpass with "electrical components", HARMONICA also offers the option of a structure made up of "physical circuits". This requires the conductor width, interaction gap and conductor length to be entered, the dimensional unit being "mm". And these values can be obtained only through the TRL85 stripline calculator, which can even be called up from the operating screen by pressing a button.

Here only the values for the first and fourth line pairs (Ze = 65 Ω, Zo = 40.8 Ω, E = 90 degrees) need be entered, with the track data (copper with 35 µm thickness and a roughness of 2 µm) and the printed circuit board and housing data (board thickness H = 0.813mm, ER = 3.38, cover height above board, HU = 13mm, TAND = 0.001).

If you then press the "Synthesis" button,

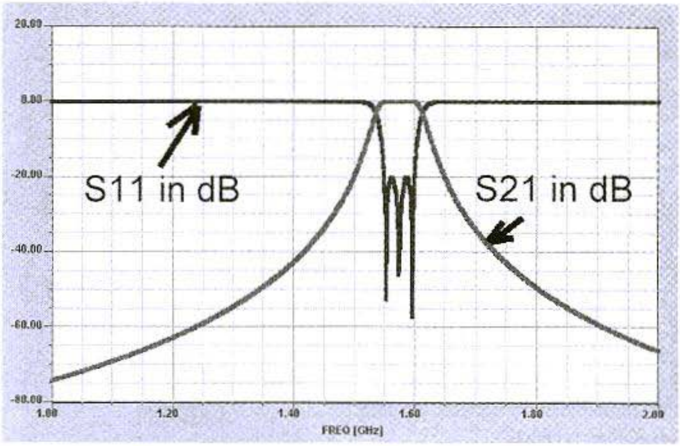


Fig 21: S11 and S21 Curves

you obtain a representation corresponding to Fig. 23, You obtain:

Conductor width W = 1.58 mm

Interaction gap width S = 0.3 mm

Circuit length P = 29.61 mm

This procedure is repeated for the second and third line pairs (Ze = 52.7 Ω, Zo = 47.56 Ω, E = 90 degrees).

Then we obtain:

W = 1.82 mm
S = 1.82 mm
P = 29.20.

It is naturally interesting to compare this with PUFF, although it should be remembered that the part involving the housing and the distance of 13 mm between the board and the cover is not included in this calculation!

Moreover, for simulation using PUFF,

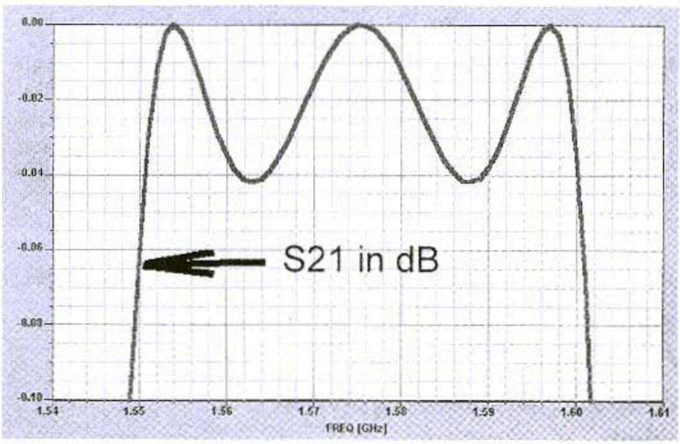
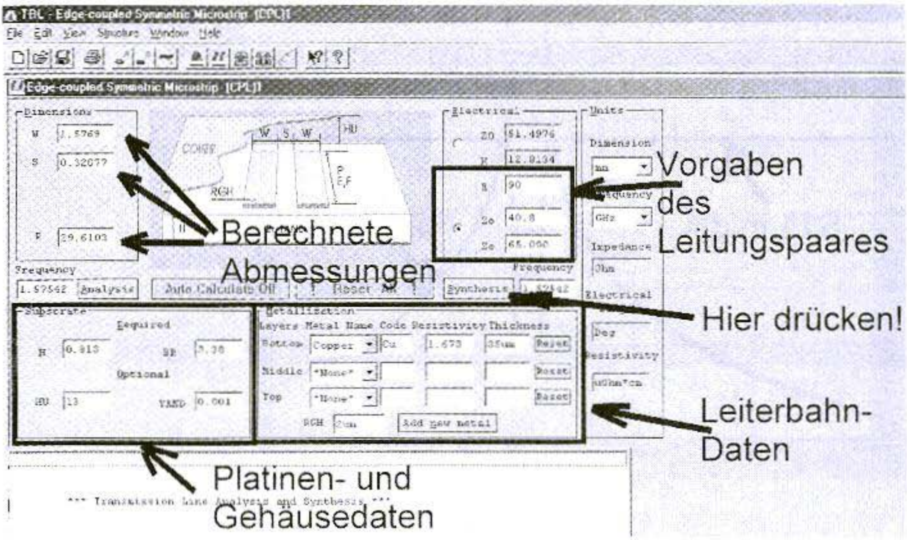


Fig 22: Cloese Up Of S21 Curve



the electrical length for the first and last line pairs is presumed to be 89 degrees, whereas for SERENADE it is 90 degrees. The uncorrected PUFF values are as follows:

Line pairs 1 + 4:
W = 1.58 mm
S = 0.31mm
Line length P = 29.34 mm;

Line pairs 2 + 3:
W = 1.82 mm
S = 1.84 mm
Line length P = 29.15 mm;

It can be seen that the differences between the two simulations are not so devastating that one of them immediately appears as completely unusable.

But let us simulate the bandpass with the "physical TRL85 values" again, consider the result and ponder on:

a) where the differences with the simulation using PUFF come from and

b) how can we arrive at the correct values.

To do this, though, we must draw a new circuit diagram, and first we must delete the old one completely.

Even when we create the new circuit diagram, things move forward splendidly. The coupled line pair is actually there with the two open ends as a completed component (Fig. 24). This naturally makes the work considerably easier.

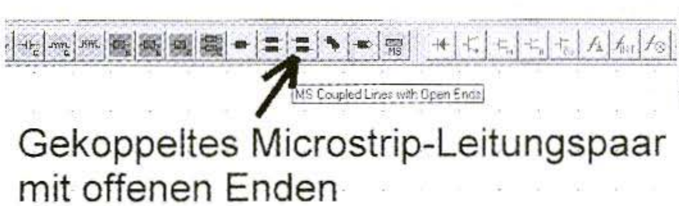


Fig 24: Component For Coupled Line Pair With Open Ends

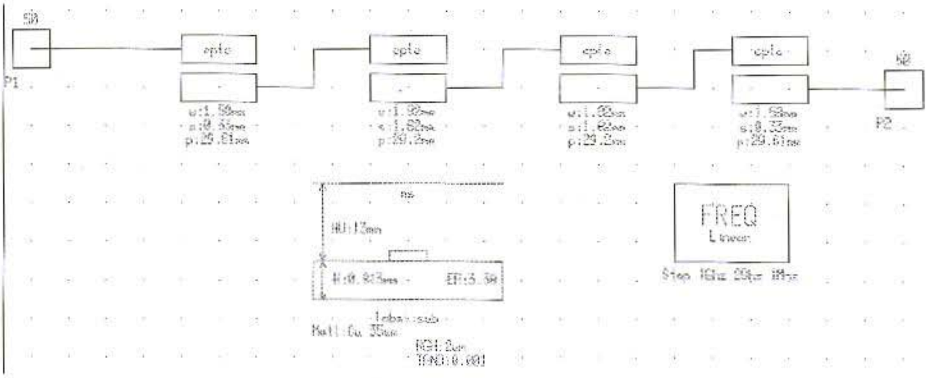


Fig 25: Revised Circuit

The screen operates in a much tidier way, even when the necessary data are entered (Fig. 25).

This is how the S-parameters look after simulation using HARMONICA (Fig. 26).

If we zoom into the representation of the passband (Fig. 27), then several points strike us:

a) The lowest transmission loss is predicted to be similar by both programmes (PUFF: approximately 2.5 dB, HARMONICA approximately 2.8 dB).

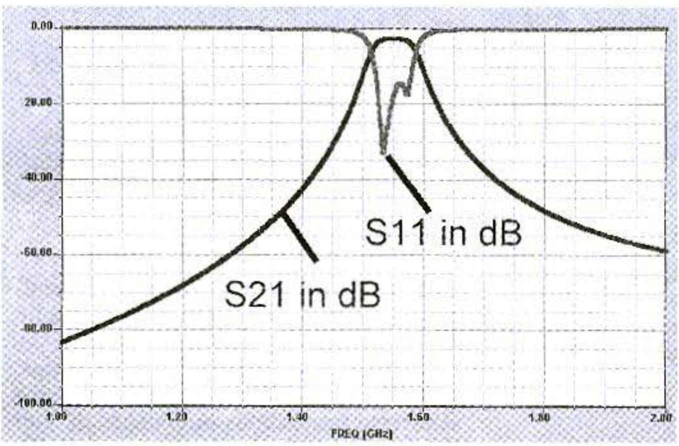


Fig 26: S Parameters From Harmonica Simulation

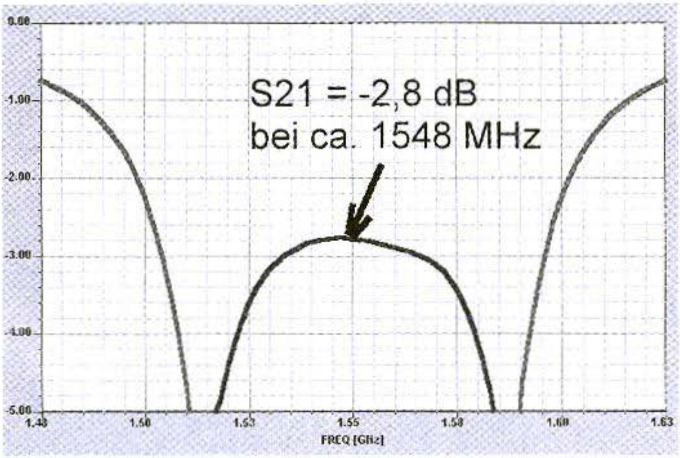


Fig 27: Close Up of S21 Curve

b) The mean frequency of the passband according to the HARMONICA simulation reveals a divergence of 30 MHz (approximately 2 %) and is clearly too low. The program therefore does not make automatic OPEN-END corrections!

Luckily, if we use Ansoft, we dont need to carry out the same actions on the diagram as are listed in the PUFF manual to make corrections here we have something which is extremely useful.

What we actually do is to take specific values for the line pair through variables, pre-set maximum and minimum values for the S-parameters at specific frequencies, and then let the optimiser do the job of reconciling all these wishes.

Here we have the following steps:

1st step:

In the first and fourth line pairs, the physical length, P, is replaced by a variable, P1. We correspondingly use variable P2 for the second and third line pairs (Fig. 28).

2nd step:

Call up a "variable control block" (Menu path: "Parts/Control Blocks/Variables") and enter the permissible variation range for P1 and P2 between question marks. In the middle is the original initial value (Fig. 29).

3rd step:

We now have to formulate the optimisation goals. There is an original button for this, in the form of a yellow and red practice target. In the student version only a maximum of 3 optimisation goals are permissible, but this should just be enough at first.

The optimisation goals here are:

a) In the "Frange1" range, from 1.55 to 1.6 GHz, S21 should not fall below 3.3 dB (Goal1)

b) In the "Frange2" range, S11 should be lower than 20 dB (Goal2) (Fig. 30).

4th step:

Only now should you press the "Optimisation" button. The program normally indicates that it must first analyse the circuit, and asks for permission to do this. Grant it permission and also finally give it precise instructions in connection with the number of optimisation searches, the

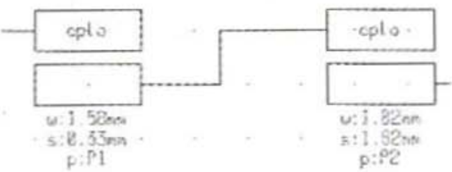


Fig 28: Variables P1 and P2

representation, etc. (Fig. 31).

5th step:

Now please follow the sequence exactly:

1) Pre-set, for example,. 2000 searches;
2) Select "RANDOM" as optimisation type;
3) This tick has to go;
4) Now press "Optimise" and wait until the program has found the best approximation to the pre-settings.
5) Now start another circuit analysis, as this is the only way to update the results diagram.
6) Now close this menu and obtain the diagrams with S11 and / or S21 in the foreground (Fig. 32).

The assessment can be found in Fig. 33, and the result looks very satisfactory.

Fig 29: Entry Of Variables P1 and P2

The data within the passband have changed in accordance with Fig. 34; the result is useable.

The only question remaining is: where have the accompanying new circuit data got to, with which the optimiser has produced the curves above?

Its very simple: youll find them in the variable block instead of the initial values (Fig. 35)!

Finally, assemble all data for the layout preparation in accordance with "ANSOFTs physical variants".

First and fourth line pairs:
- Conductor width W = 1.58 mm
- Interaction gap S = 0.33 mm
- Line length P = 29.04 mm

Second and third line pairs:
- Conductor width W = 1.82 mm
- Interaction gap S = 1.82 mm
- Line length P = 28.80 mm

Now only one question remains: "Which of the two programs is really right"?

Theres only one way to find the answer to this question: produce another printed circuit board with just these dimensions, measure it under exactly the same conditions as for the "PUFF product" using the network analyser, and then cold-blood-

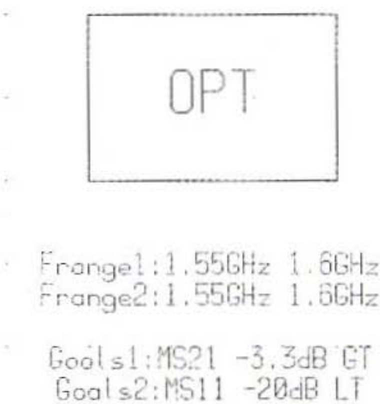


Fig 30: Goals For Optimisation

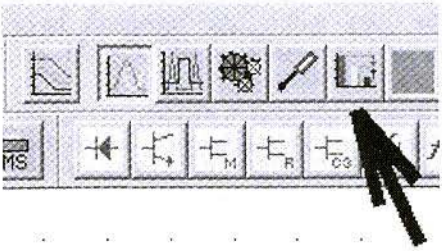


Fig 31: Starting Optimisation

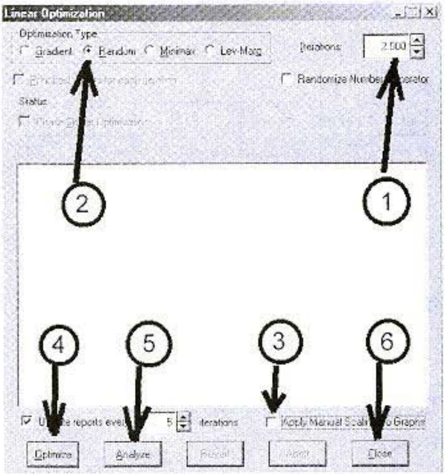


Fig 32: Closing Menu

edly analyse the results and compare them.

7. Literature

[1] Design and realisation of microwave circuits, Gunthard Kraus, VHF Communications, from issue 4/96, P. 244 - 250, at irregular intervals to issue 2/99.

[2] PUFF manual, original English version

[3] Ansoft-Serenade manuals (supplied when program is downloaded)

[4] APLAC manuals (supplied when program is downloaded)

[5] Microwave Engineering by David Pozar (John Wiley & Sons, New York, ISBN 0-471-17096-8).

[6] Microwave Filters, Impedance-Matching Networks, and Coupling Structures by G. Mattaei, L. Young and E. M. T: Jones. (Artech House Publishers,

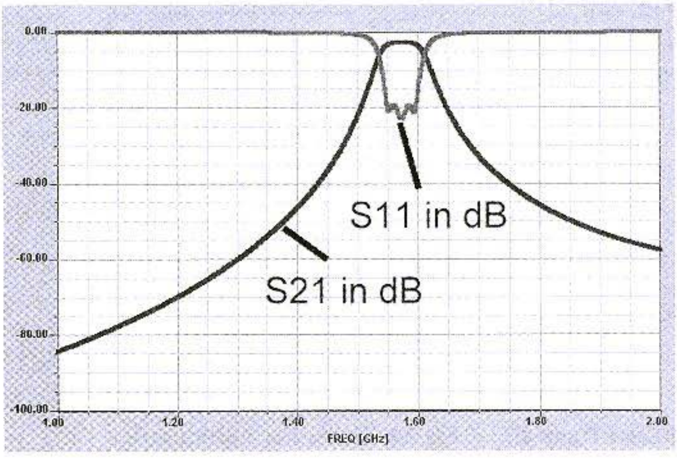


Fig 33: Simulation Results

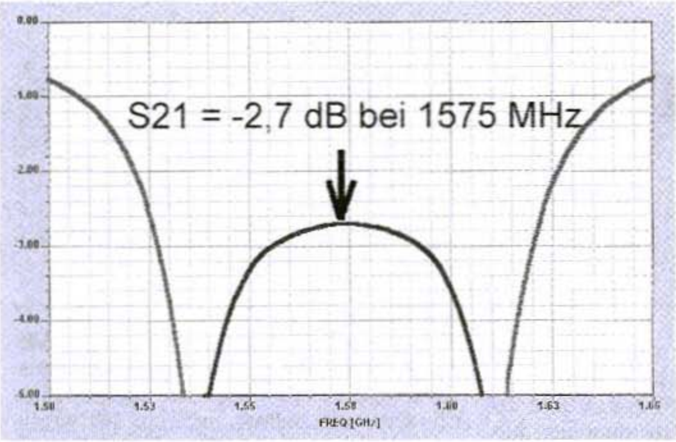


Fig 34: Simulation Results

ISBN 0-89006-099-1).

[7] Microstrip Lines and Slotlines by K: C. Gupta, Ramesh Garg, Inder Bahl and Prakash Bhartia. (Artech House Publishers, ISBN 0-89006-766-X).

Software, manuals and tutorial are combined on an "ANSOFT-CD" and can be obtained from the author, provided the costs are reimbursed. Please E-mail me at: krausg@elektronikschule.de.

Fig 35: New Circuit Data